

CFD ANALYSIS OF HEAT TRANSFER AUGMENTATION FOR FLOW THROUGH A TUBE USING WIRE COIL INSERTS

**A THESIS SUBMITTED IN PARTIAL FULFILLMENT
OF THE REQUIREMENTS FOR THE DEGREE OF**

**Bachelor of Technology
In
Chemical Engineering**

Submitted By

Shalini Patra

109CH0089

Under the guidance of

Dr. Satish Kumar Agarwal



**Department of Chemical Engineering
National Institute of Technology
Rourkela
2013**

National Institute of Technology, Rourkela



CERTIFICATE

This is to certify that the thesis entitled, “**CFD ANALYSIS OF HEAT TRANSFER AUGMENTATION FOR FLOW THROUGH A TUBE USING WIRE COIL INSERTS**”, submitted by Miss. Shalini Patra, Roll no. 109CH0089, in partial fulfillment of the requirements for the award of degree of Bachelor of Technology in Chemical Engineering at National Institute of Technology, Rourkela is an authentic work carried out by her under my supervision and guidance.

To the best of my knowledge, the matter embodied in the report has not been submitted to any other University / Institute for the award of any Degree or Diploma.

Date:

Place: Rourkela

Dr.S.K.Agarwal

Department of Chemical Engineering

National Institute of Technology

Rourkela – 769008

ACKNOWLEDGEMENT

I express my sincere gratitude to Prof. S.K.Agarwal for providing me an opportunity to work on this project and for his constant guidance and timely suggestions throughout. I am also thankful to Prof. H.M.Jena and Prof. R.K.Singh (Project Co-ordinators) for their valuable guidance and advice.

I would also like to thank M.Tech student Mr. T.Dhani Babu, and PhD scholar Mr. Akhilesh Khapre, for their help and guidance.

Finally I thank my parents, brother and friends for their support and encouragement without which this project would not have been possible.

Date:

Shalini Patra

109CH0089

Department of Chemical Engineering

National Institute of Technology, Rourkela

ABSTRACT

The need to increase the thermal performance of heat transfer equipment (for instance, heat exchangers), thereby effecting energy, material, and cost savings as well as a consequential mitigation of environmental degradation has led to the development and use of many heat transfer enhancement techniques. These methods are referred to as augmentation or intensification techniques. This project deals with the analysis of heat transfer augmentation for fluid flowing through pipes using CFD. Using CFD codes for modeling the heat and fluid flow is an efficient tool for predicting equipment performance. CFD offers a convenient means to study the detailed flows and heat exchange processes, which take place inside the tube. Friction factor and Nusselt number for water flowing through the specified pipe (internal diameter = 0.022 m, length = 2.5 m) were obtained first for the smooth pipe and then for the pipe with a wire coil insert in the Reynolds number range of 250 to 25,000 and Prandtl number of 6.97. Three wire coils with pitch 0.033 m, 0.044 m and 0.0484 m, and coil diameter 0.00154 m, 0.00187 m and 0.002 m respectively were considered. Calculated values (obtained from empirical equations available in literature) were compared with CFD values. Comparisons were also made between the smooth tube results and the coiled wire results to establish the heat transfer augmentation due to the use of insert. It was seen that the friction factor increment i.e. f_c/f_0 varied between 1.2 to 8.5 with coil 1 giving the maximum value of 8.5 at Reynolds number 2250. Similarly, the Nusselt number increment i.e. Nu_c/Nu_0 varied in the range of 1.3 to 4.9, again with coil 1 giving the maximum value of 4.9 at the same Reynolds number of 2250. Simulations were carried out using commercial CFD software ANSYS 13.0.

Keywords: Heat transfer augmentation, CFD, Friction factor, Nusselt number, wire coil inserts.

CONTENTS

<u>Chapter</u>	<u>Topic</u>	<u>Page no.</u>
	Abstract	iii
	List of figures	v
	List of tables	vi
	Nomenclature	vii
Chapter 1	Introduction	1
Chapter 2	Literature Review	4
2.1	Classification of Augmentation Techniques	5
2.2	Performance Evaluation Criteria	7
2.3	Wire Coil inserts	10
Chapter 3	Computational Fluid Dynamics	12
3.1	Definition	13
3.2	Applications and advantages of CFD	13
3.3	CFD Procedure	14
Chapter 4	Numerical and CFD investigations	17
4.1	Problem statement	18
4.2	Smooth tube	19
4.3	Tube with wire coil insert	23
Chapter 5	Results and Discussion	31
5.1	Smooth Tube Results	32
5.2	Results for tube with wire coil insert	32
Chapter 6	Conclusions	39
	References	41

LIST OF FIGURES

Fig No.	Name of figure	Page No.
2.1	Classic twisted tape insert	8
2.2	Perforated twisted tape insert	8
2.3	Notched twisted tape insert	8
2.4	Jagged twisted tape insert	8
2.5	Square cut circular ring insert	9
2.6	Micro-fins incorporated in a tube	9
2.7	Conical ring inserts	9
2.8	Wire insert	9
4.1	Geometry of smooth tube	20
4.2	Mesh generated for smooth tube	21
4.3	Geometry of tube with coiled wire	24
4.4	Mesh generated for tube with coiled wire	25
5.1	Friction factor vs Reynolds number for smooth tube (laminar flow)	32
5.2	Nusselt number vs Reynolds number for smooth tube (laminar flow)	32
5.3	Friction factor vs Reynolds number for smooth tube (turbulent flow)	33
5.4	Nusselt number vs Reynolds number for smooth tube (turbulent flow)	33
5.5	Friction factor vs Reynolds Number for Coil 1	34
5.6	Friction factor vs Reynolds Number for Coil 2	34
5.7	Friction factor vs Reynolds Number for Coil 3	35
5.8	Friction factor vs Reynolds Number (Comparison between different coils)	35
5.9	Nusselt Number vs Reynolds Number for Coil 1	36
5.10	Nusselt Number vs Reynolds Number for Coil 2	36
5.11	Nusselt Number vs Reynolds Number for Coil 3	37
5.12	Nusselt Number vs Reynolds Number (Comparison between different coils)	37

LIST OF TABLES

Table no.	Name of table	Page no.
4.1	Specifications of the wire coils used	18
4.2	Friction factor and Nusselt number for Laminar region for smooth tube	22
4.3	Friction factor and Nusselt Number for Turbulent Region for smooth tube	23
4.4	Wire coil specifications	25
4.5	Friction factor and Nusselt number for coil 1	26
4.6	Friction factor and Nusselt number for coil 2	27
4.7	Friction factor and Nusselt number for coil 3	28
4.8	Friction factor comparison of the three coils with smooth tube (based on CFD values)	29
4.9	Nusselt number comparison of the three coils with smooth tube (based on CFD values)	30

NOMENCLATURE

C_p	Specific heat of fluid, J/kg.K
D, d	Diameter of pipe, m
e	Diameter of wire coil, m
f	Fanning friction factor, Dimensionless
Gz	Graetz Number, Dimensionless
K	Thermal conductivity, W/m.K
L	Length of pipe, m
Nu	Nusselt Number, Dimensionless
Pr	Prandtl number, dimensionless
p	Pitch of wire coil, m
Re	Reynolds Number, Dimensionless
v	Flow velocity, m/s

Greek symbols:

α	Helix angle of wire coil, degrees
ρ	Density of the fluid, kg/m ³
μ	Viscosity of the fluid, N s/m ²

Subscripts:

D_i	Internal diameter of pipe, m
f_0	Friction factor for smooth tube, Dimensionless
f_c	Friction factor for tube with wire coil insert, Dimensionless
Nu_0	Nusselt Number for smooth tube, Dimensionless
Nu_c	Nusselt Number for tube with wire coil insert, Dimensionless
μ_b	Viscosity of fluid at bulk temperature, N s/m ²
μ_w	Viscosity of fluid at wall temperature, N s/m ²

CHAPTER 1

INTRODUCTION

INTRODUCTION

The conversion, utilization, and recovery of energy in industrial, commercial, and domestic application usually involve a heat transfer process. Improved heat exchange, over and above that in the usual or standard practice, can significantly improve the thermal efficiency in such applications as well as the economics of their design and operation. The need to increase the thermal performance of heat based equipments (for instance, heat exchangers), thereby effecting energy, material, and cost savings as well as a consequential mitigation of environmental degradation has led to the development and use of many heat transfer enhancement techniques. These methods are referred to as augmentation or intensification techniques.

Enhancement techniques essentially reduce, for example, the thermal resistance in a conventional heat exchanger by promoting higher convective heat transfer coefficient with or without surface area increases (as represented by fins or extended surfaces). As a result, the size of a heat exchanger can be reduced, or the heat duty of an existing exchanger can be increased, or the exchanger's operating approach temperature difference can be decreased. The latter is particularly useful in thermal processing of biochemical, food, plastic, and pharmaceutical media, to avoid thermal degradation of the end product. On the other hand, heat exchange systems in spacecraft, electronic devices, and medical applications, for example, may rely primarily on enhanced thermal performance for their successful operation.

In the present work, heat transfer enhancement for fluid flowing through a pipe with wire coil inserts is to be analyzed using Computational Fluid Dynamics (CFD).

The impressive improvements in computer performance, matched by developments in numerical methods, have resulted in a growing confidence in the ability of CFD to model complex fluid flows. CFD techniques have been applied on a broad scale in the process industry to gain insight into various flow phenomena, examine different equipment designs or compare performance under different operating conditions.

The report is divided into the following chapters:

Chapter 2 deals with the literature review that has been done with the various techniques used for enhancement and work carried out on CFD analysis of such methods.

Chapter 3 provides a general background to CFD, including its applications, advantages, CFD analysis procedure, and methodology.

Chapter 4 is regarding the present work carried out, i.e. numerical/theoretical calculation of friction factor and Nusselt number for the specified problem and comparison with values obtained from CFD using ANSYS 13.0 software.

Chapter 5 deals with the results obtained and their discussion and analysis followed by some suggestions to improve the results obtained.

Chapter 6 contains the concluding remarks.

CHAPTER 2

LITERATURE REVIEW

LITERATURE REVIEW

2.1 Classification of Augmentation Techniques:

The various enhancement techniques [1,2] can be classified broadly as *passive* and *active* techniques. Passive techniques do not require direct input of external power, unlike active techniques. They generally use surface or geometrical modifications to the flow channel, or incorporate an insert, material, or additional device. Except for extended surfaces, which increase the effective heat transfer surface area, these passive schemes promote higher heat transfer coefficients by disturbing or altering the existing flow behavior. This, however, is accompanied by an increase in the pressure drop. In the case of active techniques, the addition of external power essentially facilitates the desired flow modification and improvement in the rate of heat transfer. The use of two or more techniques (passive and/or active) in conjunction constitutes *compound augmentation techniques*.

The effectiveness of any of these methods is strongly dependent on the mode of heat transfer (single-phase free or forced convection, pool boiling, forced convection boiling or condensation, and convective mass transfer), and type and process application of the heat exchanger.

2.1.1 Passive techniques:

- *Treated surfaces* are heat transfer surfaces that have a fine-scale alteration to their finish or coating. The alteration could be continuous or discontinuous, where the roughness is much smaller than what affects single-phase heat transfer, and they are used primarily for boiling and condensing duties.
- *Rough surfaces* are generally surface modifications that promote turbulence in the flow field, primarily in single-phase flows, and do not increase the heat transfer surface area. Their geometric features range from random sand-grain roughness to discrete three-dimensional surface protuberances.
- *Extended surfaces*, more commonly referred to as finned surfaces, provide an effective heat transfer surface area enlargement. Plain fins have been used routinely in many heat exchangers. The newer developments, however, have led to modified finned surfaces that

also tend to improve the heat transfer coefficients by disturbing the flow field in addition to increasing the surface area.

- Displaced enhancement devices are inserts that are used primarily in confined forced convection, and they improve energy transport indirectly at the heat exchange surface by “displacing” the fluid from the heated or cooled surface of the duct with bulk fluid from the core flow.
- Swirl flow devices produce and superimpose swirl or secondary recirculation on the axial flow in a channel. They include helical strip or cored screw-type tube inserts, twisted ducts, and various forms of altered (tangential to axial direction) flow arrangements, and they can be used for single-phase as well as two-phase flows.
- Coiled tubes are what the name suggests, and they lead to relatively more compact heat exchangers. The tube curvature due to coiling produces secondary flows, which promote higher heat transfer coefficients in single-phase flows as well as in most regions of boiling.
- Surface tension devices consist of wicking or grooved surfaces, which direct and improve the flow of liquid to boiling surfaces and from condensing surfaces.
- Additives for liquids include the addition of solid particles, soluble trace additives, and gas bubbles in single-phase flows, and trace additives, which usually depress the surface tension of the liquid, for boiling systems.
- Additives for gases include liquid droplets or solid particles, which are introduced in single-phase gas flows in either a dilute phase (gas–solid suspensions) or dense phase (fluidized beds).

2.1.2 Active techniques:

- Mechanical aids are those that stir the fluid by mechanical mean or by rotating the surface. The more prominent examples include rotating tube heat exchangers and scraped-surface heat and mass exchangers.

- Surface vibration has been applied primarily, at either low or high frequency, in single-phase flows to obtain higher convective heat transfer coefficients.
- Fluid vibration or fluid pulsation, with vibrations ranging from 1.0 Hz to ultrasound, used primarily in single-phase flows, is considered to be perhaps the most practical type of vibration enhancement technique.
- Electrostatic fields, which could be in the form of electric or magnetic fields, or a combination of the two, from dc or ac sources, can be applied in heat exchange systems involving dielectric fluids. Depending on the application, they can promote greater bulk fluid mixing and induce forced convection (corona “wind”) or electromagnetic pumping to enhance heat transfer.
- Injection, used only in single-phase flow, pertains to the method of injecting the same or a different fluid into the main bulk fluid either through a porous heat transfer interface or upstream of the heat transfer section.
- Suction involves either vapor removal through a porous heated surface in nucleate or film boiling, or fluid withdrawal through a porous heated surface in single-phase flow.
- Jet impingement involves the direction of heating or cooling fluid perpendicularly or obliquely to the heat transfer surface. Single or multiple jets (in clusters or staged axially along the flow channel) may be used in both single-phase and boiling applications.

Furthermore, as mentioned earlier, any two or more of these techniques (passive and/or active) may be employed simultaneously to obtain enhancement in heat transfer that is greater than that produced by only one technique itself. This simultaneous utilization is termed *compound enhancement*.

2.2 Performance Evaluation Criteria:

Besides the relative thermal–hydraulic performance improvements brought about by the enhancement devices, there are many factors [2] that should be considered to evaluate the performance of particular heat transfer equipment. They include economic (engineering development, capital, installation, operating, maintenance, and other such costs),

manufacturability (machining, forming, bonding, and other production processes), reliability (material compatibility, integrity, and long-term performance), and safety, among others. The assessment of these factors, as well as the enhanced convection performance, is usually application driven. In most practical applications of enhancement techniques, the following performance objectives, along with a set of operating constraints and conditions, are usually considered for optimizing the use of a heat exchanger:

1. Increase the heat duty of an existing heat exchanger without altering the pumping power (or pressure drop) or flow rate requirements.
2. Reduce the approach temperature difference between the two heat-exchanging fluid streams for a specified heat load and size of exchanger.
3. Reduce the size or heat transfer surface area requirements for a specified heat duty and pressure drop.
4. Reduce the process stream's pumping power requirements for a given heat load and exchanger surface area.

It may be noted that objectives 1, 2, and 4 yield savings in operating (or energy) costs, and objective 3 lends to material savings and reduced capital costs. These objective functions and constraints have been described by many different performance evaluation criteria (PEC) in the literature.

Below are some of the inserts [4] used for augmentation commonly:-



Fig 2.1 Classic twisted tape



Fig 2.2 Perforated twisted tape



Fig 2.3 Notched twisted tape



Fig 2.4 Jagged twisted tape



Fig 2.5 Square cut circular ring insert

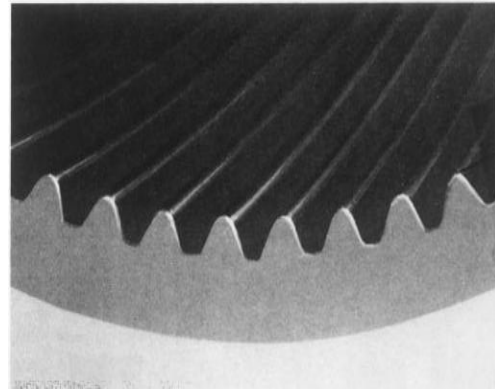


Fig 2.6 Micro-fins incorporated in a tube

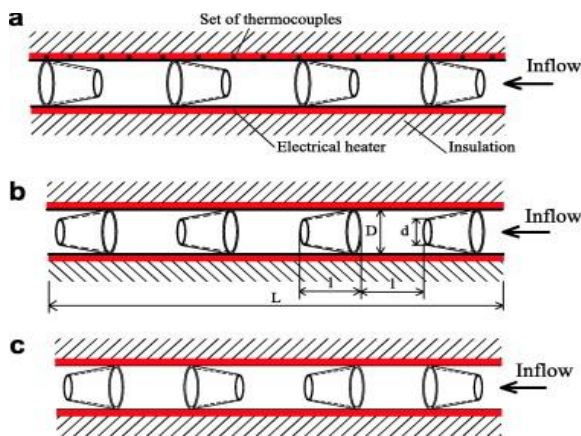


Fig 2.7 Conical ring inserts

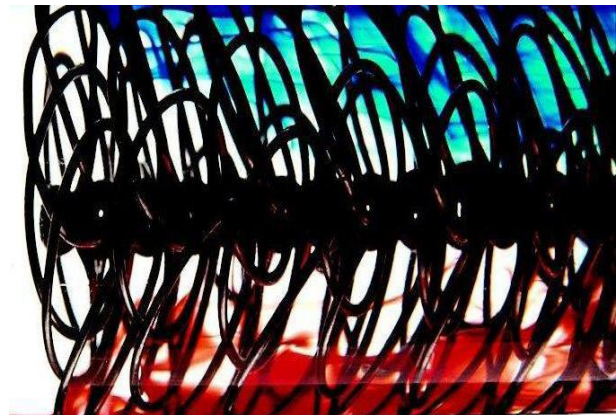


Fig 2.8 Wire insert

The quick development of numerical techniques and software, such as CFD codes, has improved the ability of engineers to solve highly sophisticated engineering problems. Using CFD codes for modeling the heat and fluid flow is an efficient tool for predicting equipments performance. The CFD offers a convenient means to study the detailed flows and heat exchange processes, which take place inside the tube. Many efforts have been undertaken to use the CFD modeling for designing heat transfer enhancement devices.

Eiamsa-ard et al. [3] carried out an experimental investigation on heat transfer and friction factor characteristics in a double-pipe heat exchanger fitted with regularly spaced twisted tape insert. Two types of tube inserts consisting of a full-length typical twisted tape at different twisted ratios and a twisted tape with various free space ratios were used in their experiments. Their results showed that the heat transfer coefficient increased by decreasing the twist ratio. However, they illustrated that decreasing the free space ratio can increase both the heat transfer coefficient and friction factor.

Rahimi et al. [4] carried out a CFD modeling in order to find the heat transfer in a tube equipped with different modified inserts fabricated based on a twisted tape insert. The operational performances of these inserts were compared experimentally and theoretically with that obtained from the classic twisted tape insert. The CFD predicted results have been used to explain the experimental observations.

S. Eiamsa-ard et al. [5] reported the details of the turbulence modeling to help in understanding of the behaviors of the incompressible swirl flows for tube fitted with the loose-fit twisted tapes in comparison with those for a tube equipped with tight-fit twisted tapes. In this work, the standard $k-\epsilon$ turbulence model, the Renormalized Group (RNG) $k-\epsilon$ turbulence model, the standard $k-\omega$ turbulence model, and the Shear Stress Transport (SST) $k-\omega$ turbulence model, were performed to study the phenomena of flow field (velocity vector and streamline), temperature field, pressure field and turbulent intensity (TKE) in a tube with twisted tape inserts. Similar works were carried out with different inserts. Heat transfer enhancement in a channel due to the presence of a triangular prism was obtained using numerical simulation [6]. The order of enhancement was about 15%. However, as expected, the augmentation was associated with enhanced skin friction.

Zhang et al. [7] investigated the heat transfer characteristics of a helically baffled heat exchanger combined with a finned tube experimentally and theoretically. Commercial Fluent 6.0 CFD code was used for predicting its fluid flow and heat transfer performances. The authors reported a good agreement between the modeling and experimental results.

2.3 Wire Coil inserts:

Garcia, Vicente and Viedma [8] carried out experiments with six different helical wire coils fitted inside a round tube to establish their thermohydraulic performance in laminar, transition and turbulent flow. Experimental correlations of Fanning friction factor and Nusselt number as functions of flow and system parameters were proposed. It was seen that wire coil inserts offer their best performance within the transition region where they show a considerable advantage over other enhancement techniques.

Kumar et al. [9] experimentally investigated the heat transfer enhancement by wire coil inserts inside a double pipe heat exchanger with engine oil flowing in the inner tube and steam in the annulus. Two empirical correlations were developed for predicting heat transfer enhancement of these inserts in the laminar region.

Numerical simulations of laminar flow in pipes with coil wire inserts were carried out by Esparza and Rojas [10]. Three-dimensional numerical simulations of the incompressible laminar flow in smooth round pipes of diameter, d , with wire coil inserts of helical pitch, p , and diameter, e , were accomplished with the finite volume method. The effect of pitch on the friction factor was addressed by performing a parametrical study with a pitch-periodic computational domain showing that the increase of the non-dimensional pitch, p/d , decreases the friction factor.

CHAPTER 3

COMPUTATIONAL FLUID DYNAMICS

COMPUTATIONAL FLUID DYNAMICS

3.1 Definition:

[11] Computational Fluid Dynamics (CFD) is the use of computer-based simulation to analyze systems involving fluid flow, heat transfer and associated phenomena such as chemical reaction. A numerical model is first constructed using a set of mathematical equations that describe the flow. These equations are then solved using a computer programme in order to obtain the flow variables throughout the flow domain.

Since the advent of the digital computer, CFD has received extensive attention and has been widely used to study various aspects of fluid dynamics. The development and application of CFD have undergone considerable growth, and as a result it has become a powerful tool in the design and analysis of engineering and other processes. In the early 1980s, computers became sufficiently powerful for general-purpose CFD software to become available.

3.2 Applications and advantages of CFD:

Many developments in the field of CFD along with its ability to model complex phenomena and growing popularity of such softwares have widened the range of application of CFD. It is applied in a wide range of industries including mechanical, process, petroleum, power, metallurgical, biomedical, pharmaceutical and food industries. CFD techniques have been applied on a broad scale in the process industry to gain insight into various flow phenomena, examine different equipment designs or compare performance under different operating conditions. Examples of CFD applications in the chemical process industry include drying, combustion, separation, heat exchange, mass transfer, pipeline flow, reaction, mixing, multiphase systems and material processing. CFD has also been applied to a number of food processing operations such as drying, refrigeration, sterilization, mixing and heat exchangers. CFD has also been successfully used in modelling various multiphase flow systems, particularly gas-solid mixtures, although some limitations still exist. Multiphase CFD models can help understand the complex interactions

between the different phases and provide detailed 3-D transient information that experimental approaches may not be able to provide. These applications, amongst others, demonstrate the potential of CFD to simulate complex flows and therefore the possibility of utilizing it to investigate a wider range of processes.

3.3 CFD procedure:

All commercial CFD packages involve sophisticated user interfaces to input parameters and to examine the results. Hence all the codes consist of three main elements:

- Pre-processor
- Solver
- Post-processor

Pre-processing is the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The steps involved in this are:

- Definition of geometry.
- Grid generation.
- Selection of the phenomena or system to be modeled.
- Definition of fluid properties.
- Boundary conditions specification.

Solver involves the following steps:

- Approximation of the unknown flow variables by means of simple functions.
- Discretization by substitution of the approximations into the governing flow equations.
- Solution of algebraic equations.

Post-processor- All the leading CFD packages are equipped with versatile data visualization tools. These include:-

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- Color postscript output

The mathematical modelling of a flow problem is achieved basically through three steps:

- developing the governing equations describing the flow;
- discretization of the governing equations; and

- solving the resulting numerical equations.

Governing equations:

- *Unsteady state 3-D equation of continuity:-*

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0$$

where ρ is the fluid density, t is time, and \mathbf{u} is velocity.

- *Momentum equation:-*

$$\frac{\partial}{\partial t}(\rho u_i) + \text{div}(\rho u_i \mathbf{u}) = -\frac{\partial P}{\partial x_i} + \text{div}(\mu \text{grad } u_i)$$

where P is pressure, μ is fluid viscosity, x is the coordinate and the subscript i indicates the Cartesian coordinates.

- *Energy equation:-*

$$\frac{\partial}{\partial t}(\rho i) + \text{div}(\rho i \mathbf{u}) = -P \text{div } \mathbf{u} + \text{div}(k \text{grad } T) + \Phi + S_i$$

where k is the thermal conductivity, T is the temperature, Φ is the dissipation term and S is the source term.

The governing equations shown above are partial differential equations (PDEs). Since digital computers can only recognize and manipulate numerical data, these equations cannot be solved directly. Therefore, the PDEs must be transformed into numerical equations containing only numbers and no derivatives. This process of producing a numerical analogue to the PDEs is called ‘numerical discretization’. Various techniques used for discretization are the finite difference method, the finite element method, and the finite volume method.

- *Finite Difference method* describes the unknowns of the flow problem by means of point samples at the node points of a grid co-ordinate lines. Taylor series expansions are used to generate finite difference approximations of derivatives in terms of point samples at each grid point and its immediate neighbors. Those derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation.
- *Finite Element Method* uses piecewise functions valid on elements to describe the local variations of unknown flow variables. Here also a set of algebraic equations are generated to determine unknown co-efficients.

- *Finite Volume method* is probably the most popular method used for numerical discretization in CFD. This method is similar in some ways to the finite difference method. This approach involves the discretisation of the spatial domain into finite control volumes. The governing equations in their differential form are integrated over each control volume. The resulting integral conservation laws are exactly satisfied for each control volume and for the entire domain, which is a distinct advantage of the finite volume method. Each integral term is then converted into a discrete form, thus yielding discretised equations at the centroids, or nodal points, of the control volumes.

The commercial CFD software used for the current project is ANSYS 13.0.

CHAPTER 4

NUMERICAL AND CFD INVESTIGATIONS

4.1 Problem statement:

(a) A smooth copper pipe 2.5 m in length and 0.022 m internal diameter is considered. Water flows through it with inlet temperature 25°C. Wall temperature is assumed constant at 100°C. Properties of water at 25°C are as follows:

Density, $\rho = 995.7 \text{ kg/m}^3$

Thermal conductivity, $K = 0.618 \text{ W/m.K}$

Viscosity, $\mu = 801.2 \times 10^{-6} \text{ kg/m.s}$

Specific heat, $C_p = 4.174 \text{ kJ/kg.K}$

Nusselt number and friction factor values were calculated from empirical equations taking different velocity values in the laminar and turbulent regions. These were then compared with the values obtained from CFD simulation.

(b) Same procedure was repeated for pipe with wire coil insert in it. Three different wires were considered for the calculations and simulations with different pitch and wire diameters as shown below:

Table 4.1 Specifications of the wire coils used:

Coil no.	Pitch p(m)	Coil diameter e (m)	Tube diameter d (m)	p/d	e/d	p/e	p^2/ed
1	0.0330	0.00154	0.022	1.5	0.0700	21.43	32.143
2	0.0440	0.00187	0.022	2.0	0.0850	23.53	47.059
3	0.0484	0.00200	0.022	2.2	0.0909	24.20	53.240

4.2 Smooth tube:

4.2(a) Equations used:-

- *Reynold's number*, $Re = \rho v D_i / \mu$
- *Friction factor*, f :
 - Laminar flow: $f = 16/Re$
 - Turbulent flow: $f = 0.079(Re)^{-0.25}$ (Blassius equation)
- *Nusselt Number*, Nu :
 - Laminar flow ($Re < 2100$):

For $Gz < 100$, Hausen's equation is used:

$$Nu = 3.66 + [0.085Gz / (1 + 0.047Gz^{2/3})](\mu_b/\mu_w)^{0.14}$$

where $Gz = Re.Pr.(D/L)$

$$Pr = C_p \mu / K$$

For $Gz > 100$, Sieder Tate equation is used:

$$Nu = 1.86Gz^{1/3}(\mu_b/\mu_w)^{0.14}$$

- Transition region ($2100 < Re < 10000$):

$$Nu = 0.116(Re^{2/3} - 125)Pr^{1/3}[1 + (D/L)^{2/3}](\mu_b/\mu_w)^{0.14}$$

- Turbulent flow ($Re > 10000$):

Sieder Tate equation is used:

$$Nu = 0.023Re^{0.8}Pr^{1/3}(\mu_b/\mu_w)^{0.14}$$

4.2(b) CFD Modelling:

Geometry:

The geometry i.e smooth cylindrical smooth tube of required dimensions was created using ANSYS Design Modeler as shown below:-

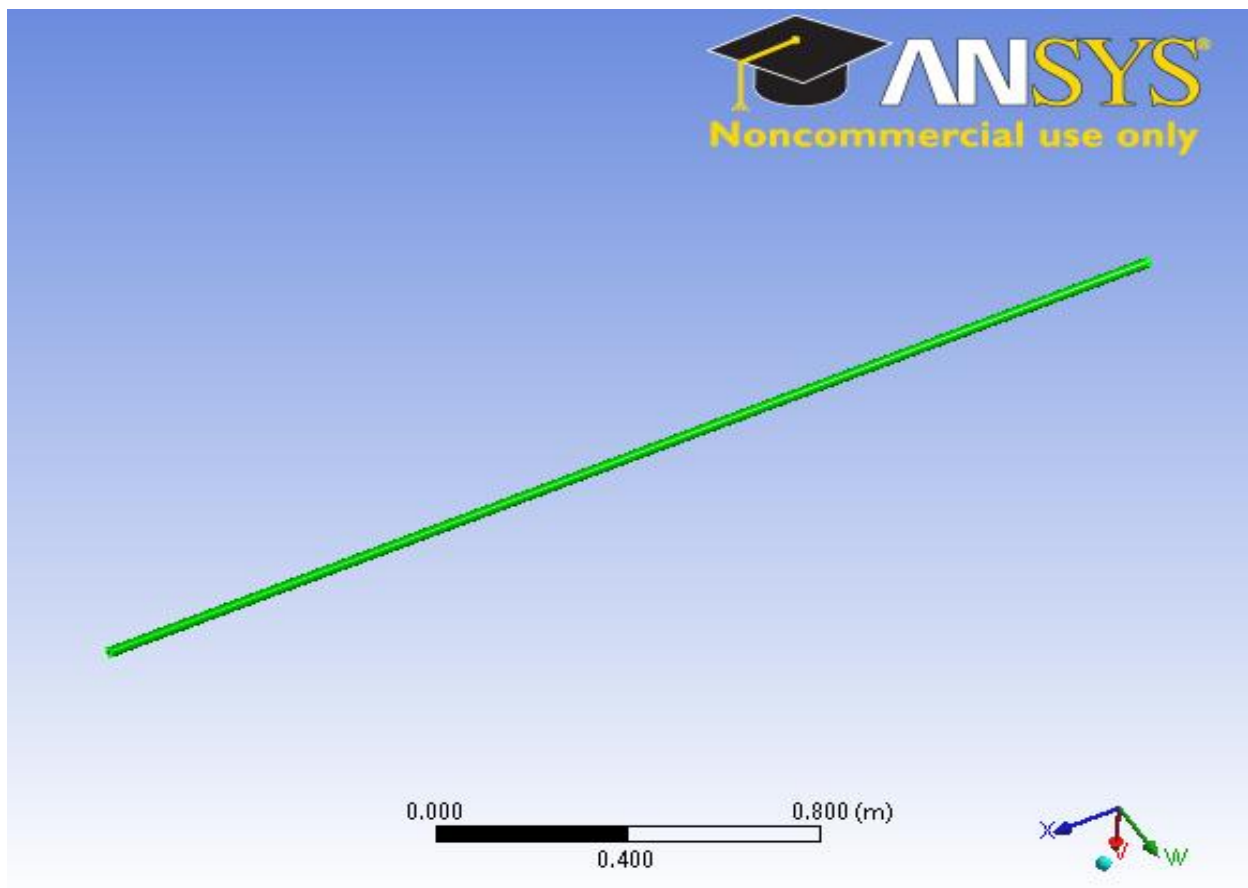


Fig 4.1 Geometry of smooth tube

Meshing:

After creation of geometry, meshing was done. A section of the mesh generated is shown below:-

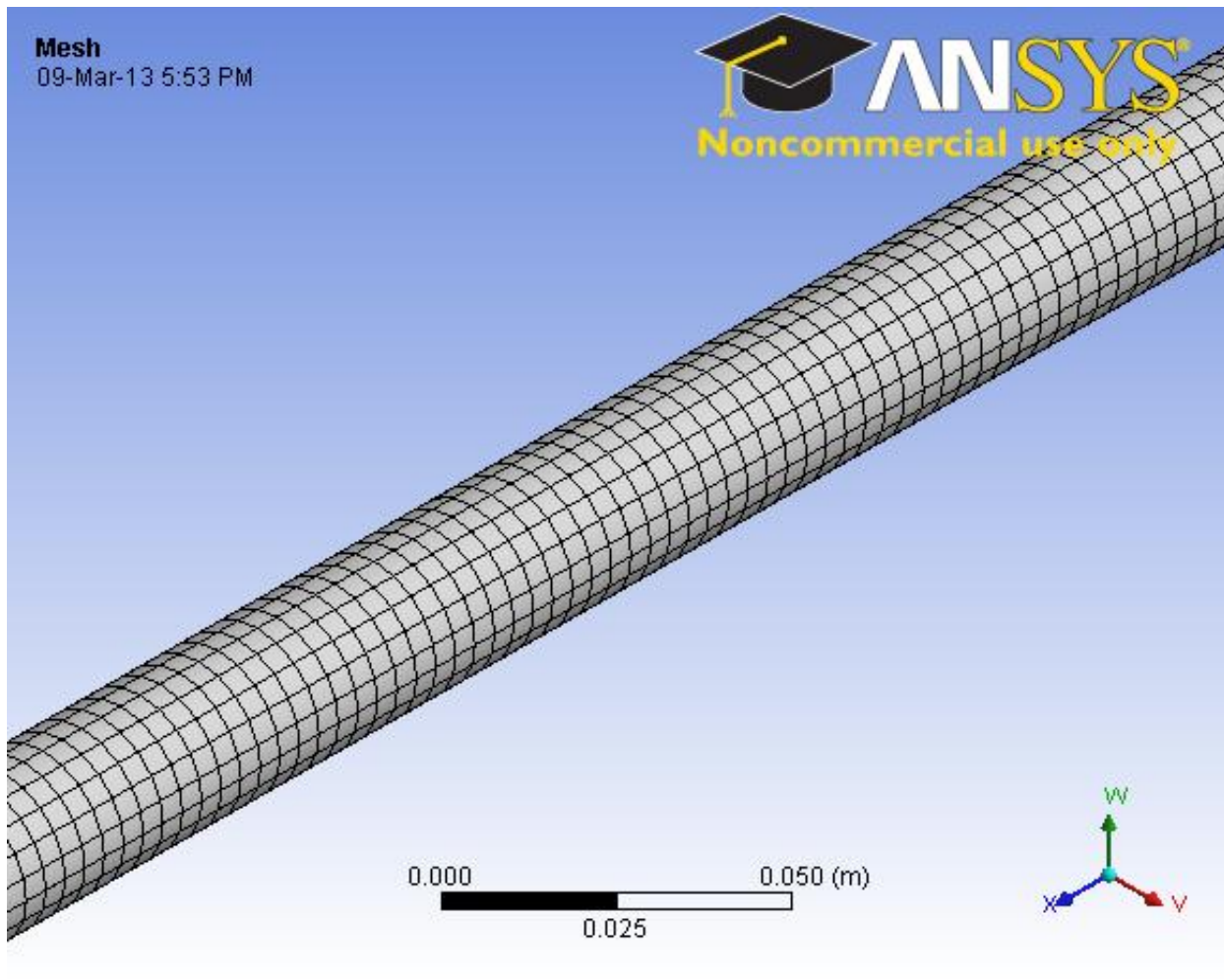


Fig 4.2 Mesh generated for smooth tube

For laminar flow, the viscous (laminar) model was used whereas the k- ϵ turbulence model with enhanced wall treatment was used for turbulent flow. The SIMPLE scheme with first order UPWIND method was used as the solution method.

4.2(c) Tabulation:

Table 4.2 Friction factor and Nusselt number for Laminar region for smooth tube:

v(m/s)	Re	f(calc)	f(CFD)	%diff	Nu(calc)	Nu(CFD)	%diff
0.01	250	0.064	0.0659	-2.96875	4.785562	3.595173	24.8746
0.015	375	0.042667	0.0439	-2.89063	5.231271	4.546115	13.09731
0.02	500	0.032	0.0329	-2.8125	5.632918	5.328742	5.399964
0.025	625	0.0256	0.0263	-2.73438	6.000755	6.016348	-0.25987
0.03	750	0.021333	0.0219	-2.65625	6.341427	6.608824	-4.21668
0.035	875	0.018286	0.0188	-2.8125	6.659594	7.175464	-7.74626
0.04	1000	0.016	0.0165	-3.125	6.958697	7.706479	-10.746
0.045	1125	0.014222	0.0146	-2.65625	7.241372	8.250873	-13.9407
0.05	1250	0.0128	0.0132	-3.125	7.509697	8.715061	-16.0508
0.055	1375	0.011636	0.0119	-2.26563	7.765349	9.148839	-17.8162
0.06	1500	0.010667	0.0109	-2.1875	8.762357	9.554552	-9.04088
0.065	1625	0.009846	0.0101	-2.57812	8.999292	9.934233	-10.3891
0.07	1750	0.009143	0.0094	-2.8125	9.224367	10.28982	-11.5504
0.075	1875	0.008533	0.0088	-3.125	9.438963	10.623	-12.5441
0.08	2000	0.008	0.0082	-2.5	9.644222	10.93544	-13.3885
0.085	2125	0.007529	0.0077	-2.26563	9.841097	11.22859	-14.099

Basis of % diff calculation: calculated values,

e.g. for friction factor, % diff = $[(f(\text{calc})-f(\text{CFD}))/f(\text{calc})]*100$

Similarly, for Nusselt number, % diff = $[(Nu(\text{calc})-Nu(\text{CFD}))/Nu(\text{calc})]*100$

Table 4.3 Friction factor and Nusselt Number for Turbulent Region for smooth tube:

v(m/s)	Re	f(calc)	f(CFD)	%diff	Nu(calc)	Nu(CFD)	%diff
0.1	2500	0.012124	0.01509	-24.4639	22.96981	23.62	-2.83063
0.2	5000	0.009655	0.01111	-15.0699	39.99277	41.63	-4.09381
0.3	7500	0.008545	0.00966	-13.0486	55.31647	58.23	-5.26702
0.4	10000	0.00787	0.0088	-11.817	69.63145	73.9	-6.1302
0.5	12500	0.0074	0.00811	-9.59459	83.24026	88.59	-6.42687
0.6	15000	0.007046	0.00759	-7.72069	96.31156	102.43	-6.35276
0.7	17500	0.006767	0.00725	-7.13758	108.9522	115.75	-6.23925
0.8	20000	0.006538	0.00694	-6.14867	121.2354	129.03	-6.42931
0.9	22500	0.006345	0.0067	-5.59496	133.2145	142.29	-6.8127
1	25000	0.00618	0.00648	-4.85437	144.9297	155.35	-7.1899

4.3 Tube with coiled wire insert:

4.3(a) Equations used [8,9]:-

- *Friction factor, f:*

- Laminar flow:

$$f = 16.8/(\text{Re})^{0.96}$$

- Turbulent flow:

$$f = 9.35(p/e)^{-1.16}(\text{Re})^{-0.217}$$

- *Nusselt Number, Nu:*

- Laminar flow:

$$\text{Nu} = 0.91(\tan\alpha)\text{Re}^{[0.29(\tan\alpha)^{-0.21}]} \text{Pr}^{0.33}(\mu_b/\mu_w)^{0.14} \quad (1.0 < p/d < 1.8)$$

$$\text{Nu} = 1.63(\tan\alpha)\text{Re}^{[0.26(\tan\alpha)^{-0.37}]} \text{Pr}^{0.33}(\mu_b/\mu_w)^{0.14} \quad (1.8 < p/d < 2.65)$$

Where α is the helix angle in degrees.

- Turbulent flow:

$$Nu = 0.132(p/d)^{-0.372} Re^{0.72} Pr^{0.37}$$

4.3(b) CFD Modelling:

Geometry:

Following geometry was created in the ANSYS Design Modeler. The coil was drawn by specifying the pitch in twist specification option under “sweep” operation.

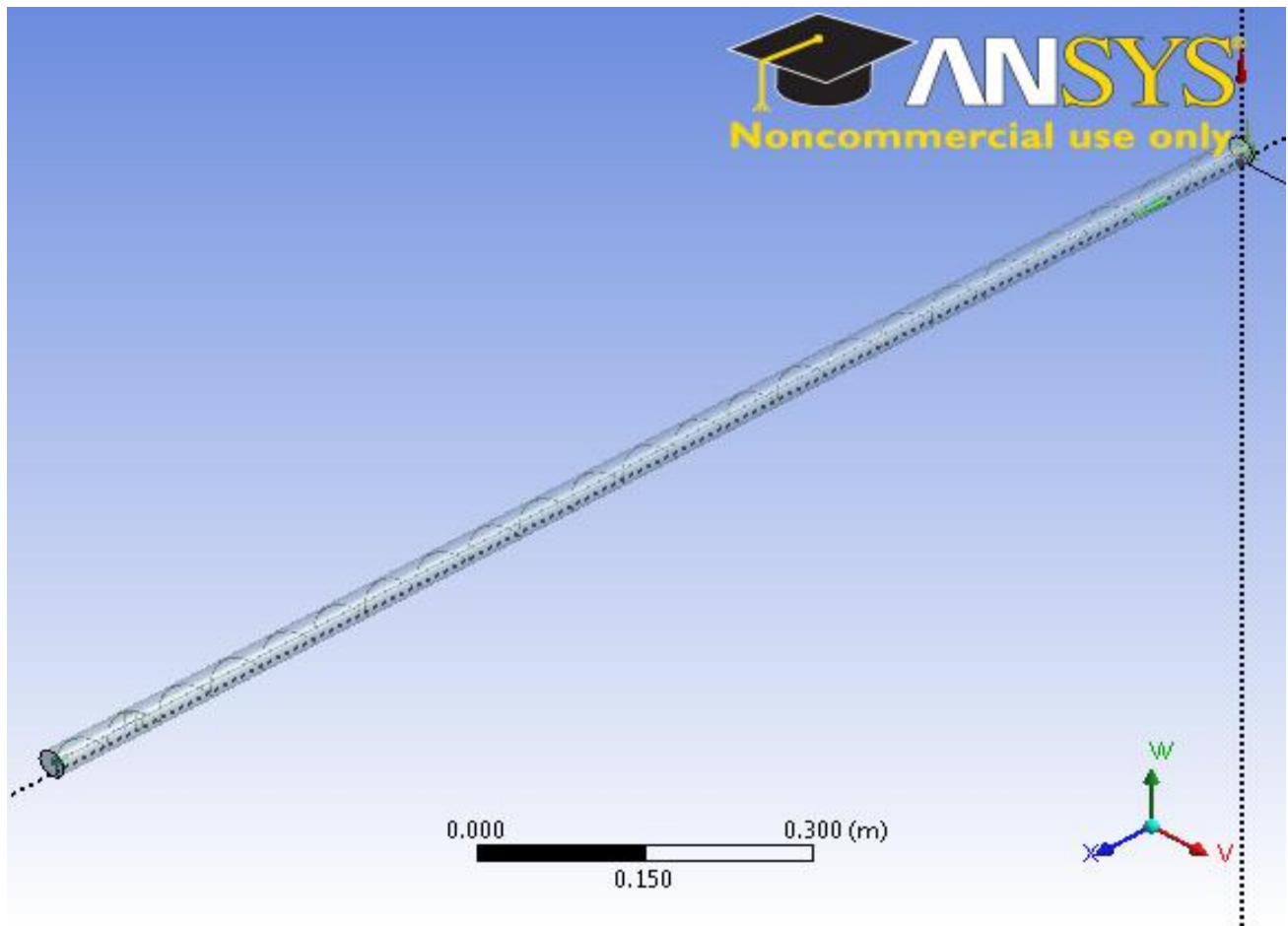


Fig 4.3 Geometry of tube with coiled wire

Meshing:

Tetrahedron, patch independent mesh was generated as shown below:-

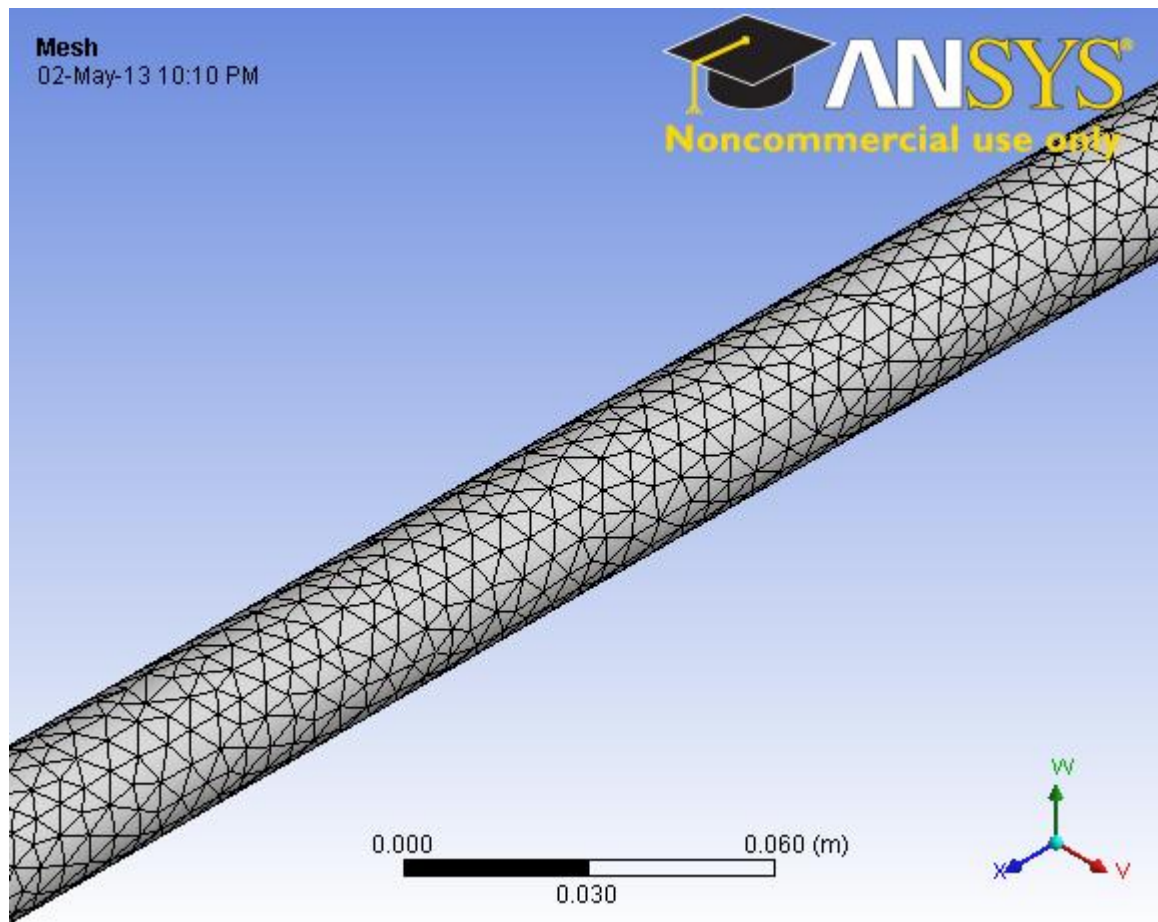


Fig 4.4 Mesh generated for tube with coiled wire

Three such geometries were generated with the various pitch and diameter specifications given below. The models used for simulation were the same as that in case of smooth tube.

Table 4.4 Wire coil specifications

Coil no.	Pitch p(m)	Coil diameter e (m)	Tube diameter d (m)	p/d	e/d	p/e	p^2/ed
1	0.0330	0.00154	0.022	1.5	0.0700	21.43	32.143
2	0.0440	0.00187	0.022	2.0	0.0850	23.53	47.059
3	0.0484	0.00200	0.022	2.2	0.0909	24.20	53.240

4.2(c) Tabulation:

Table 4.5 Friction factor and Nusselt number for coil 1:

v(m/s)	Re	$f_{cl}(\text{calc})$	$f_{cl}(\text{CFD})$	%diff	$Nu_{cl}(\text{calc})$	$Nu_{cl}(\text{CFD})$	%diff
0.01	250	0.083808	0.08225	1.859327	6.586951	4.73	28.19136
0.02	500	0.043082	0.04568	-6.02982	8.194244	8.51	-3.85338
0.03	750	0.029191	0.03215	-10.1363	9.3106	11.08	-19.0041
0.04	1000	0.022147	0.02494	-12.6126	10.19374	12.85	-26.0578
0.05	1250	0.017876	0.02042	-14.23	39.51465	36.7928	6.88215
0.06	1500	0.015006	0.01731	-15.3547	45.05767	43.94604	2.467113
0.07	1750	0.012942	0.01504	-16.2131	50.34663	45.61573	9.396643
0.08	2000	0.051347	0.06045	-17.729	55.42741	46.94639	15.30112
0.09	2250	0.050051	0.0596	-19.0785	60.33293	58.03086	3.815608
0.1	2500	0.04892	0.05803	-18.6231	65.08783	73.13027	-12.3563
0.2	5000	0.042088	0.04715	-12.0267	107.2115	121.1613	-13.0114
0.3	7500	0.038543	0.04185	-8.57937	143.5579	163.5772	-13.9451
0.4	10000	0.036211	0.03635	-0.3848	176.5969	202.6457	-14.7504
0.5	12500	0.034499	0.03275	5.069816	207.3759	238.9396	-15.2205
0.6	15000	0.033161	0.03015	9.079322	236.466	273.1727	-15.523
0.7	17500	0.03207	0.0282	12.067	264.2228	305.9435	-15.7899
0.8	20000	0.031154	0.0266	14.61753	290.8872	336.9805	-15.8458
0.9	22500	0.030368	0.02525	16.85263	316.6317	366.4965	-15.7485
1	25000	0.029681	0.02415	18.63574	341.5858	395.7205	-15.8481

Table 4.6 Friction factor and Nusselt number for coil 2:

v(m/s)	Re	$f_{c2}(\text{calc})$	$f_{c2}(\text{CFD})$	%diff	$Nu_{c2}(\text{calc})$	$Nu_{c2}(\text{CFD})$	%diff
0.01	250	0.083808	0.084101	-0.34929	9.890042	7.5542	23.61812
0.02	500	0.043082	0.046859	-8.76645	12.48373	11.3678	8.939047
0.03	750	0.029191	0.03018	-3.38767	14.30576	13.6817	4.362291
0.04	1000	0.022147	0.024485	-10.5582	15.75761	14.8056	6.041581
0.05	1250	0.017876	0.019196	-7.38289	35.50887	28.62404	19.38906
0.06	1500	0.015006	0.014493	3.417905	40.48996	35.45233	12.44169
0.07	1750	0.012942	0.013381	-3.39411	45.24276	41.86387	7.468349
0.08	2000	0.046062	0.047524	-3.17357	49.80848	42.98521	13.69902
0.09	2250	0.0449	0.046587	-3.7577	54.2167	43.89676	19.03462
0.1	2500	0.043885	0.04475	-1.97131	58.48958	67.19429	-14.8825
0.2	5000	0.037756	0.039582	-4.83497	96.34298	111.9157	-16.1638
0.3	7500	0.034576	0.035448	-2.5208	129.0048	151.5947	-17.5109
0.4	10000	0.032484	0.033639	-3.55595	158.6944	187.7474	-18.3075
0.5	12500	0.030948	0.031436	-1.57544	186.3532	221.5045	-18.8627
0.6	15000	0.029748	0.030893	-3.84937	212.4943	253.7522	-19.416
0.7	17500	0.028769	0.029336	-1.96993	237.4374	283.9901	-19.6063
0.8	20000	0.027948	0.028596	-2.32006	261.3986	312.6507	-19.6069
0.9	22500	0.027242	0.028003	-2.79219	284.5333	341.1351	-19.8928
1	25000	0.026627	0.027343	-2.6907	306.9577	370.1271	-20.5792

Table 4.7 Friction factor and Nusselt number for coil 3:

v(m/s)	Re	$f_{c3}(\text{calc})$	$f_{c3}(\text{CFD})$	%diff	$Nu_{c3}(\text{calc})$	$Nu_{c3}(\text{CFD})$	%diff
0.01	250	0.083808	0.086341	-3.02205	9.60436	7.220761	24.81789
0.02	500	0.043082	0.043685	-1.39914	12.22438	13.04487	-6.71188
0.03	750	0.029191	0.029899	-2.42505	14.07689	17.06022	-21.1931
0.04	1000	0.022147	0.025063	-13.168	15.55913	19.85513	-27.6108
0.05	1250	0.017876	0.018659	-4.3789	34.2698	28.13174	17.91099
0.06	1500	0.015006	0.016531	-10.1634	39.07707	35.09291	10.19566
0.07	1750	0.012942	0.015112	-16.7694	43.66402	41.61837	4.684975
0.08	2000	0.044602	0.043343	2.82214	48.07042	42.83706	10.88687
0.09	2250	0.043476	0.042246	2.829585	52.32482	43.83198	16.231
0.1	2500	0.042493	0.041071	3.347493	56.44859	63.53851	-12.5599
0.2	5000	0.036559	0.036433	0.345643	92.98111	105.7519	-13.7348
0.3	7500	0.03348	0.033003	1.425049	124.5032	143.2044	-15.0207
0.4	10000	0.031454	0.031192	0.832792	153.1568	177.5037	-15.8967
0.5	12500	0.029967	0.029001	3.224082	179.8504	209.1767	-16.3059
0.6	15000	0.028805	0.028036	2.668658	205.0794	239.1566	-16.6166
0.7	17500	0.027857	0.027143	2.563436	229.152	267.7214	-16.8313
0.8	20000	0.027061	0.026493	2.100716	252.2771	294.535	-16.7506
0.9	22500	0.026379	0.025412	3.664286	274.6046	320.5969	-16.7486
1	25000	0.025782	0.024928	3.313626	296.2464	346.6331	-17.0084

Table 4.8 Friction factor comparison of the three coils with smooth tube (based on CFD values):

v(m/s)	Re	f_0	f_{c1}	f_{c2}	f_{c3}	f_{c1}/f_0	f_{c2}/f_0	f_{c3}/f_0
0.01	250	0.0659	0.08225	0.084101	0.086341	1.248103	1.276191	1.310182
0.02	500	0.0329	0.04568	0.046859	0.043685	1.38845	1.424286	1.327812
0.03	750	0.0219	0.03215	0.03018	0.029899	1.468037	1.378082	1.365251
0.04	1000	0.0165	0.02494	0.024485	0.025063	1.511515	1.483939	1.51897
0.05	1250	0.0132	0.02042	0.019196	0.018659	1.54697	1.454242	1.413561
0.06	1500	0.0109	0.01731	0.014493	0.016531	1.588073	1.329633	1.516606
0.07	1750	0.0094	0.01504	0.013381	0.015112	1.6	1.423511	1.60766
0.08	2000	0.0082	0.06045	0.047524	0.043343	7.371951	5.79561	5.285732
0.09	2250	0.007	0.0596	0.046587	0.042246	8.514286	6.655286	6.035143
0.1	2500	0.01509	0.05803	0.04475	0.041071	3.845593	2.96554	2.721736
0.2	5000	0.01111	0.04715	0.039582	0.036433	4.243924	3.562736	3.279298
0.3	7500	0.00966	0.04185	0.035448	0.033003	4.332298	3.669565	3.41646
0.4	10000	0.0088	0.03635	0.033639	0.031192	4.130682	3.822614	3.544545
0.5	12500	0.00811	0.03275	0.031436	0.029001	4.038224	3.876202	3.575956
0.6	15000	0.00759	0.03015	0.030893	0.028036	3.972332	4.070224	3.693808
0.7	17500	0.00725	0.0282	0.029336	0.027143	3.889655	4.046345	3.743862
0.8	20000	0.00694	0.0266	0.028596	0.026493	3.832853	4.120461	3.817435
0.9	22500	0.0067	0.02525	0.028003	0.025412	3.768657	4.179552	3.792836
1	25000	0.00648	0.02415	0.027343	0.024928	3.726852	4.219599	3.846914

Table 4.9 Nusselt number comparison of the three coils with smooth tube (based on CFD values):

v(m/s)	Re	Nu ₀	Nu _{c1}	Nu _{c2}	Nu _{c3}	Nu _{c1} /Nu ₀	Nu _{c2} /Nu ₀	Nu _{c3} /Nu ₀
0.01	250	3.595	4.73	7.5542	7.220761	1.315716	2.101307	2.008557
0.02	500	5.329	8.51	11.3678	13.04487	1.596922	2.133196	2.447901
0.03	750	6.609	11.08	13.6817	17.06022	1.676502	2.070162	2.581361
0.04	1000	7.706	12.85	14.8056	19.85513	1.667532	1.921308	2.576581
0.05	1250	8.715	36.7928	28.62404	28.13174	4.221779	3.284456	3.227967
0.06	1500	9.555	43.94604	35.45233	35.09291	4.599272	3.710343	3.672727
0.07	1750	10.29	45.61573	41.86387	41.61837	4.433016	4.068403	4.044545
0.08	2000	10.935	46.94639	42.98521	42.83706	4.293223	3.930975	3.917426
0.09	2250	11.859	58.03086	43.89676	43.83198	4.893402	3.701556	3.696094
0.1	2500	23.62	73.13027	67.19429	63.53851	3.096116	2.844805	2.69003
0.2	5000	41.63	121.1613	111.9157	105.7519	2.910431	2.688343	2.54028
0.3	7500	58.23	163.5772	151.5947	143.2044	2.809156	2.603378	2.45929
0.4	10000	73.9	202.6457	187.7474	177.5037	2.742161	2.540561	2.401945
0.5	12500	88.59	238.9396	221.5045	209.1767	2.697139	2.500333	2.361177
0.6	15000	102.43	273.1727	253.7522	239.1566	2.66692	2.477323	2.33483
0.7	17500	115.75	305.9435	283.9901	267.7214	2.64314	2.453478	2.312927
0.8	20000	129.03	336.9805	312.6507	294.535	2.611645	2.423085	2.282686
0.9	22500	142.29	366.4965	341.1351	320.5969	2.575701	2.397463	2.253123
1	25000	155.35	395.7205	370.1271	346.6331	2.547284	2.382537	2.231304

CHAPTER 5

RESULTS AND DISCUSSION

RESULTS AND DISCUSSION

5.1 Smooth Tube Results:

5.1(a) Laminar region:

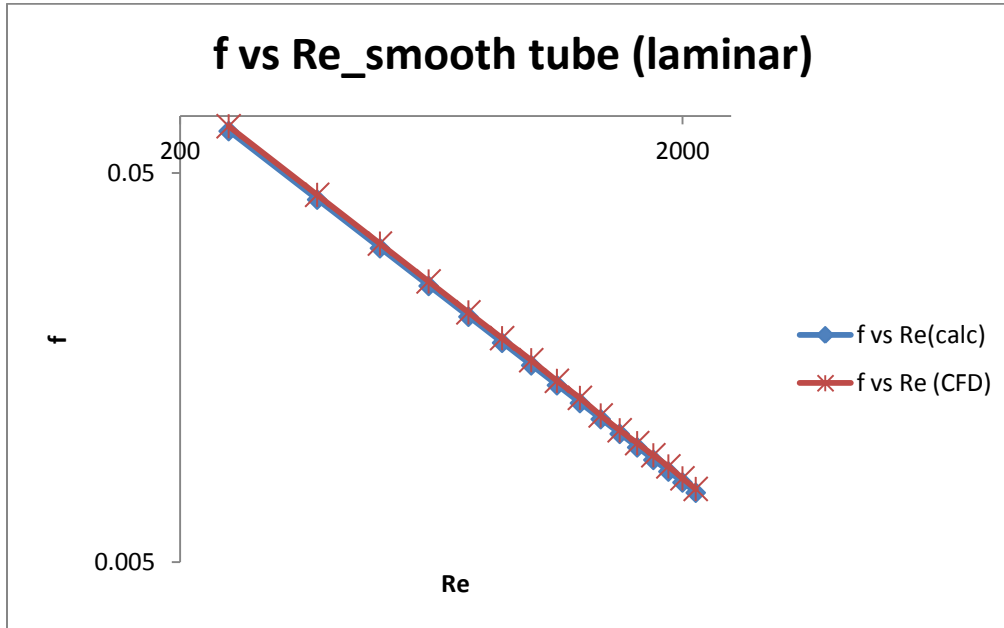


Fig 5.1 Friction factor vs Reynolds number for smooth tube (laminar flow)

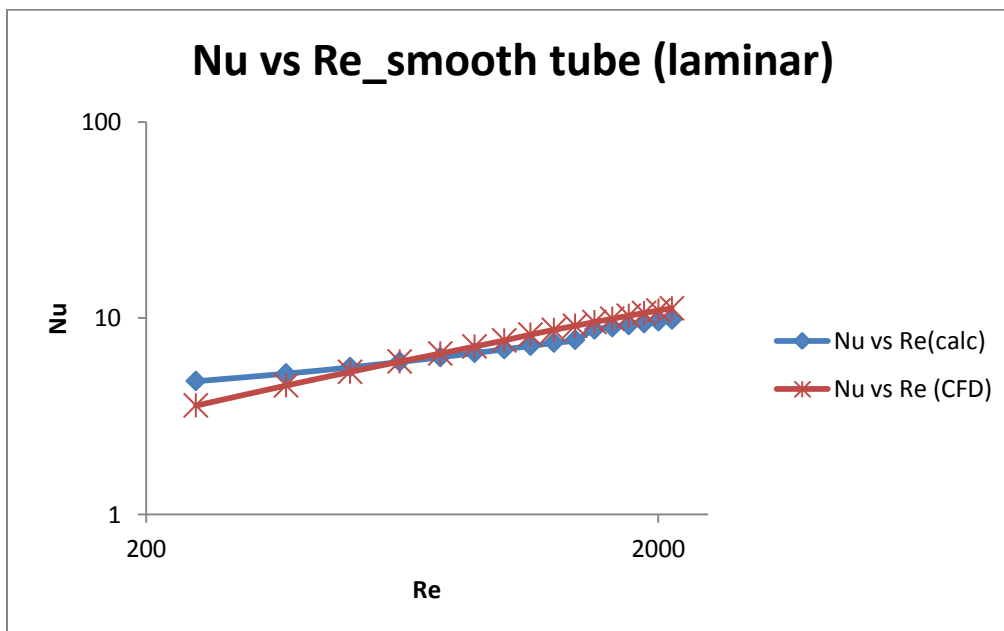


Fig 5.2 Nusselt number vs Reynolds number for smooth tube (laminar flow)

5.1(b) Turbulent Region:

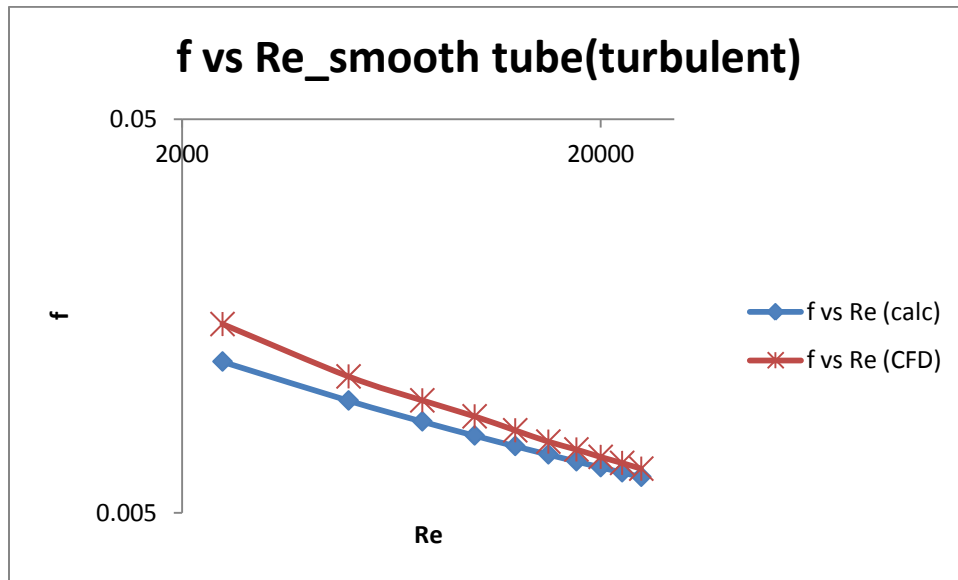


Fig 5.3 Friction factor vs Reynolds number for smooth tube (turbulent flow)

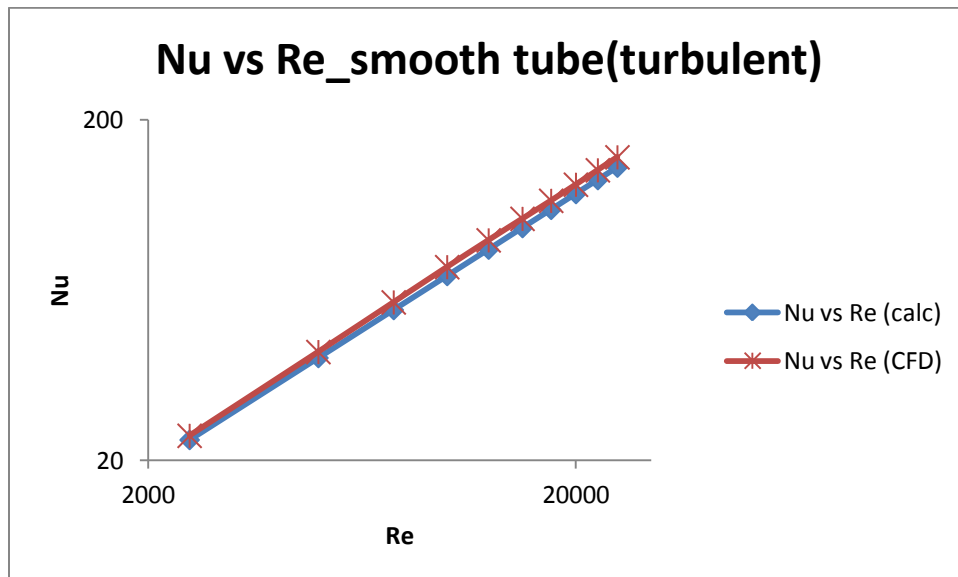


Fig 5.4 Nusselt number vs Reynolds number for smooth tube (turbulent flow)

Thus, as can be seen from the plots, the calculated values and CFD values are more or less in agreement with each other except for some deviations ranging from -3.125 to -2.18% for friction factor and -14.1 to 24.8% for Nusselt number. These deviations can be further narrowed down by increasing the number of elements during meshing. However that would lead to higher

computational time. It was also seen that changing the solution method during simulation didn't affect the result much. So, using the same models and methods, simulations for the tube with wire coil insert were carried out.

5.2 Results for tube with wire coil insert:

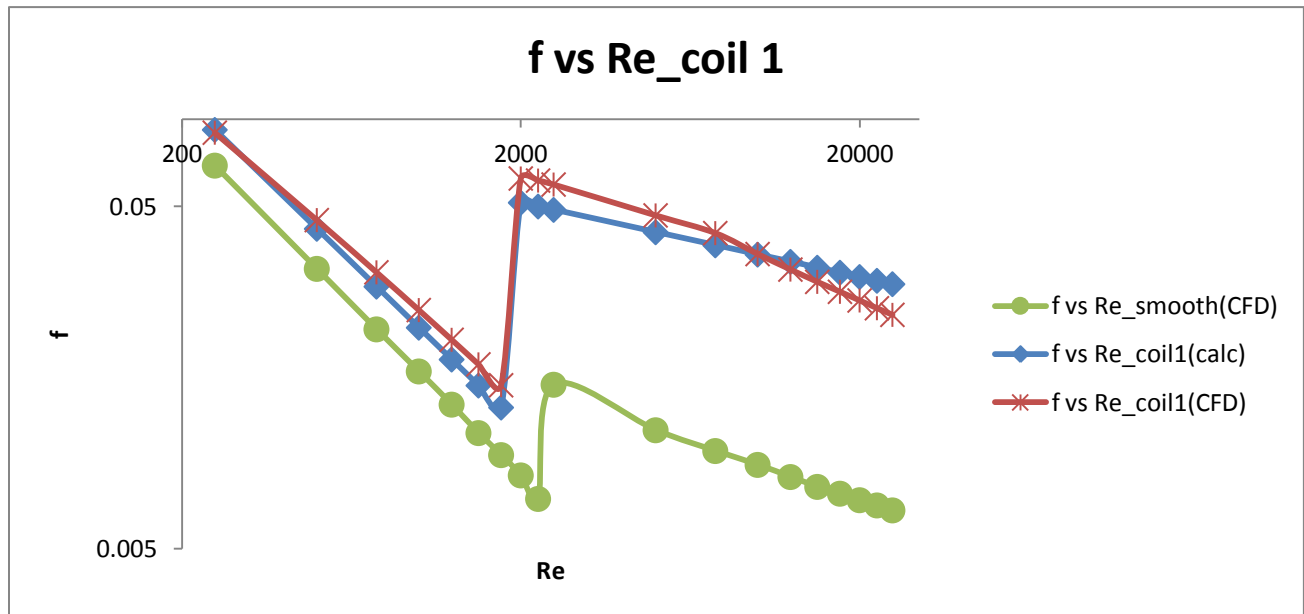


Fig 5.5 Friction factor vs Reynolds Number for Coil 1

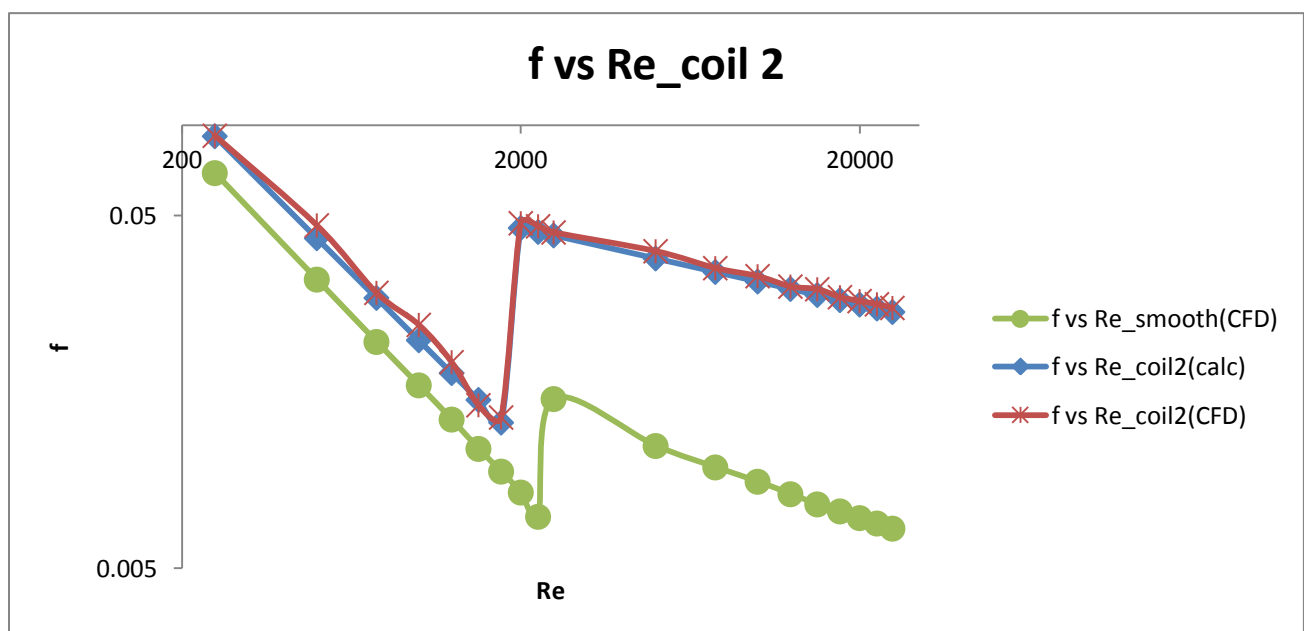


Fig 5.6 Friction factor vs Reynolds Number for Coil 2

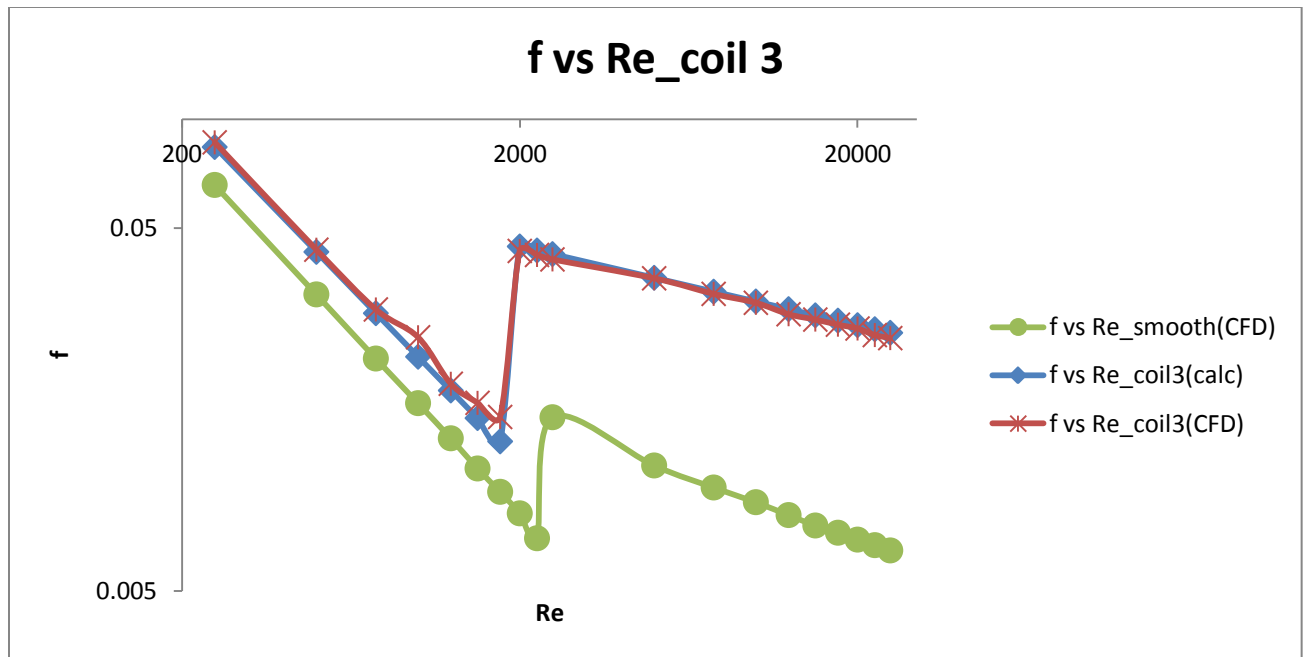


Fig 5.7 Friction factor vs Reynolds Number for Coil 3

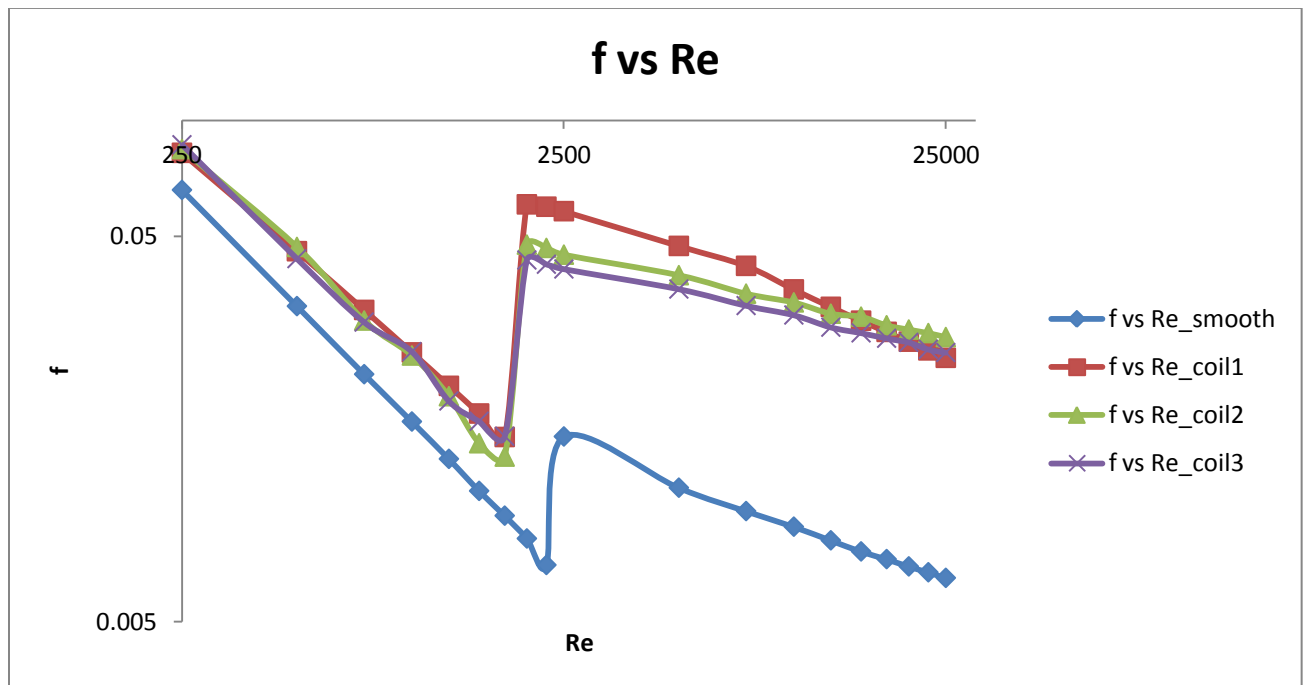


Fig 5.8 Friction factor vs Reynolds Number (Comparison between different coils)

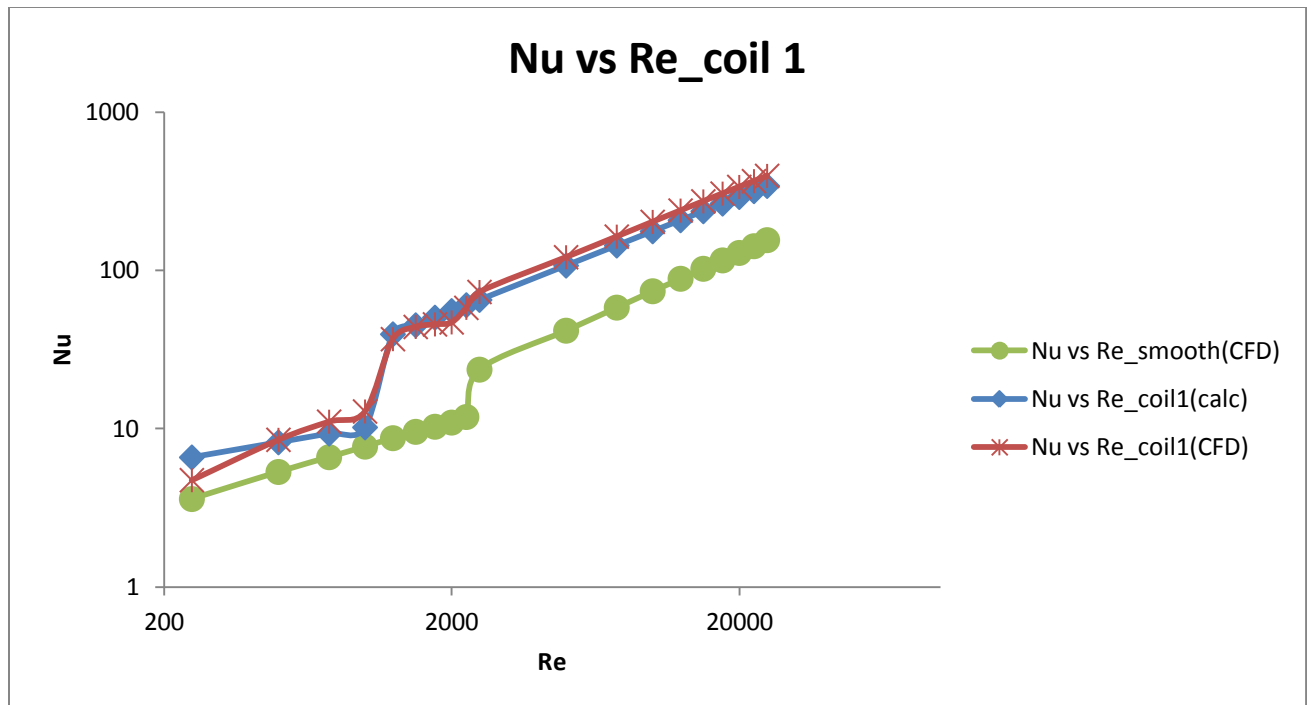


Fig 5.9 Nusselt Number vs Reynolds Number for Coil 1

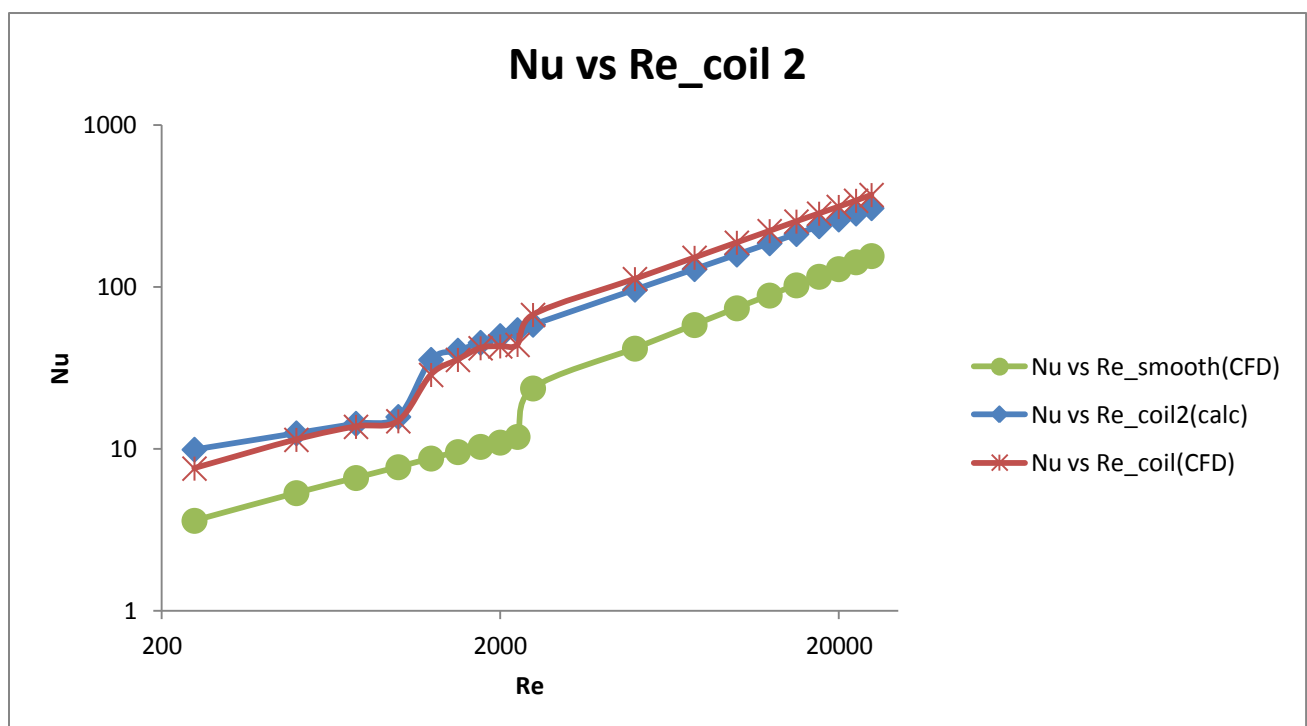


Fig 5.10 Nusselt Number vs Reynolds Number for Coil 2

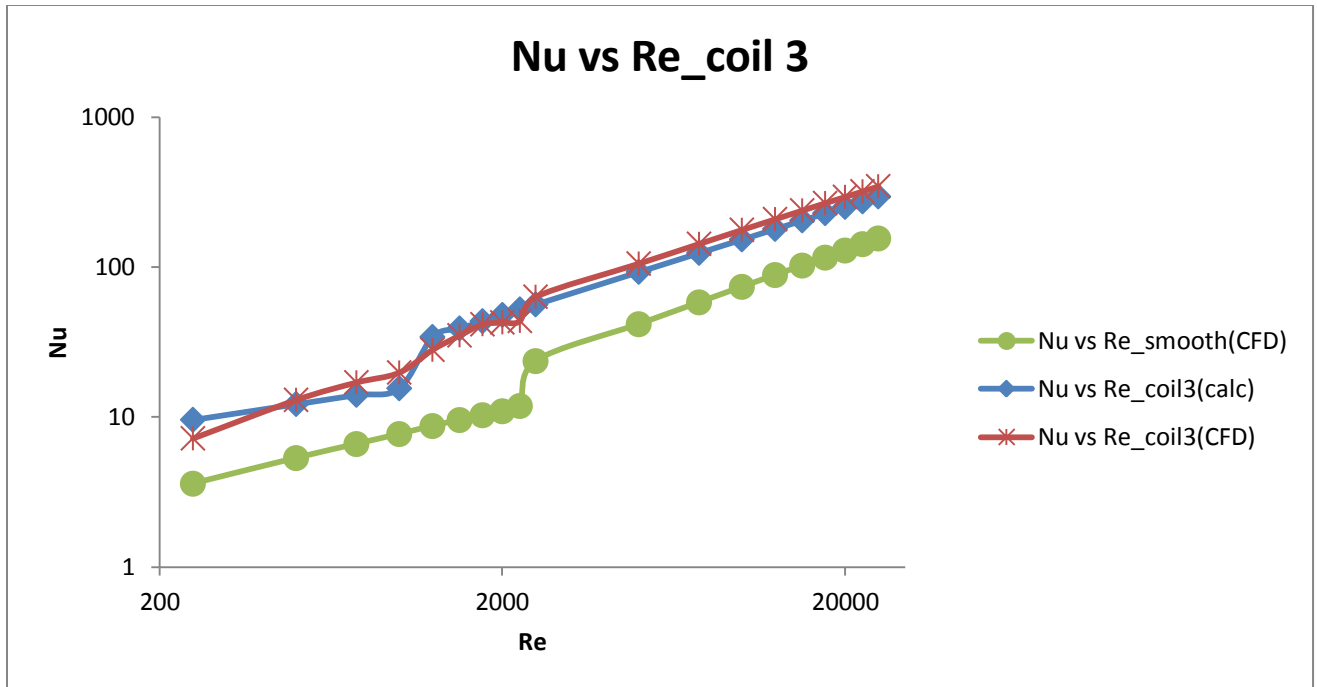
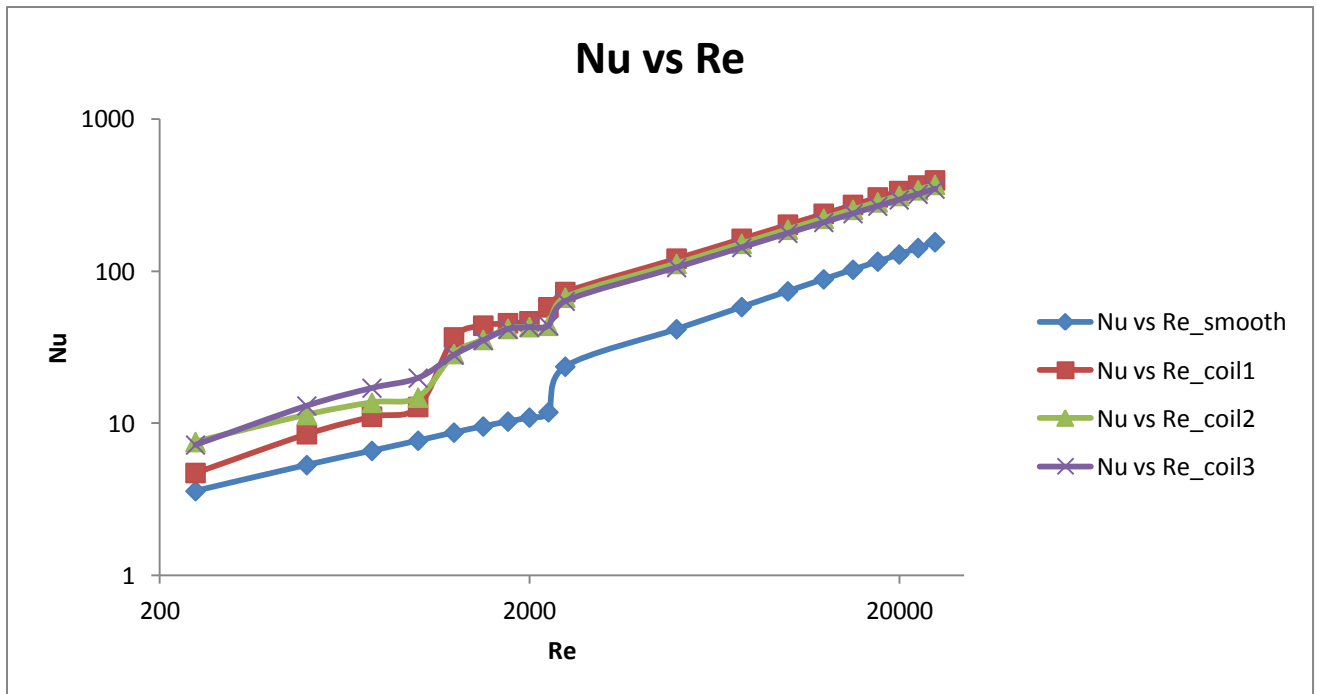


Fig 5.11 Nusselt Number vs Reynolds Number for Coil 3



5.12 Nusselt Number vs Reynolds Number (Comparison between different coils)

The CFD values follow the same trend as the values obtained from the empirical equations available in literature with deviations varying in the range of $\pm 18\%$ for friction factor and -20 to 28 % for Nusselt number. The deviation of the CFD values from the calculated ones can be due to a few reasons. Firstly, the correlations used for calculation are not universal laws or formulae, rather they are the empirical correlations developed by a particular group of researchers based on their experimental investigations. So, the values calculated may not be the true values. Secondly, CFD calculates the values by iteratively solving the discretized energy and momentum equations by the finite volume method. So, the accuracy depends on the number of iterations. Increasing the number of iterations would increase the accuracy. Another method to get better results would be to go for higher order meshing. However, increasing the number of elements during meshing can lead to very high computation time. Moreover, the commercial CFD software used for the current work did not simulate the model for very high order meshing. Hence, overcoming these problems could lead to even better results.

As can be seen, the wire coils provide moderate friction factor increase for pure laminar flow but in the transition and turbulent regions, much higher increment was observed. The increment is maximum in the transition region and again drops as Re increases and flow becomes more turbulent. Similar observations were made for Nusselt number also. The plots also indicate an early onset of turbulence at Reynolds number around 750 to 1000 due to the use of wire coils. Higher the pitch, lower is the roughness and higher the coil diameter, higher is the roughness. The wire coils used are in increasing order of pitch as well as coil diameter. This leads to counter acting effect on roughness because of which there's not much difference in the increment of friction factor and Nusselt number for the three coils. The dimensionless number p^2/ed is a measure degree of roughness. Lower its value, higher is the roughness. Thus, coil 1 having the lowest p^2/ed value i.e. 32.143 has highest roughness and hence, as expected the friction factor values for coil 1 are slightly higher than the other coils.

CHAPTER 6

CONCLUSIONS

CONCLUSION

Heat transfer analysis for water flowing through a smooth tube as well as a tube with a wire coil insert was done by calculation of friction factor and Nusselt number at the specified conditions using the empirical equations available in literature. CFD simulations were carried out for the same problem using commercial CFD software ANSYS 13.0. Results revealed that in laminar flow, wire coils mostly behave as a smooth tube with moderate increase in friction factor and Nusselt number values. However in turbulent flow, considerable increase in friction factor and Nusselt number are observed, especially for coil 1 with lowest p^2/ed ratio, i.e highest degree of surface roughness. Moreover use of wire coils gives the advantage of early onset of turbulence (at Reynolds number around 750 to 1000). It was seen that the friction factor increment i.e. f_c/f_0 varied from 1.2 to 8.5 with coil 1 giving the maximum value of 8.5 at a Reynolds number of 2250. Similarly, the Nusselt number increment i.e. Nu_c/Nu_0 varied in the range of 1.3 to 4.9, again with coil 1 giving the maximum value of 4.9 at the same Reynolds number of 2250.

Scope for future work:

Further detailed studies can be carried out in this area either through experiments or with the aid of softwares. Nusselt number and friction factor values can be obtained for wire coils with the same pitch at different coil diameters and similarly for coils with the same diameter and different pitch in order to study the effect of coil diameter and pitch respectively on Nusselt number and friction factor. Some other inserts may be used and similar investigations can be done and the values compared to those of wire coil inserts.

REFERENCES

- [1] Bergles, A.E. Techniques to augment heat transfer, Handbook of Heat Transfer Applications (Ed.W.M. Rosenhow), 1985, Ch.3 (McGraw-Hill, New York).
- [2] Bergles, A.E., Bunn R.L and Junkhan G.H, Extended performance evaluation criteria for enhanced heat transfer surfaces, Letters in Heat and Mass Transfer, Volume 1, Issue 2, November–December 1974, Pages 113-120.
- [3] S. Eiamsa-ard, C. Thianpong, P. Promvonge, Experimental investigation of heat transfer and flow friction in a circular tube fitted with regularly spaced twisted tape elements, Int. Commun. Heat Mass Transfer 33 (2006) 1225–1233.
- [4] Rahimi M, Shabanian S.R. and Alsairafi A.A, Experimental and CFD studies on heat transfer and friction factor characteristics of a tube equipped with modified twisted tape inserts, Chemical Engineering and Processing 48 (2009) 762–770.
- [5] Eiamsa-ard S, Wongcharee K, and Sripattanapipat S, 3-D Numerical simulation of swirling flow and convective heat transfer in a circular tube induced by means of loose-fit twisted tapes, International Communications in Heat and Mass Transfer 36 (2009) 947–955.
- [6] Chattopadhyay H, Augmentation of heat transfer in a channel using a triangular prism, International Journal of Thermal Sciences 46 (2007) 501–505.
- [7] Zhang Z, Ma D, Fang X, and Gao X, Experimental and numerical heat transfer in a helically baffled heat exchanger combined with one three-dimensional finned tube, Chem. Eng. Process. 47 (2008) 1738–1743.
- [8] Garcia A, Vicente P.G, and Viedma A, Experimental study of heat transfer enhancement with wire coil inserts in laminar-transition-turbulent regimes at different Prandtl numbers, International Journal of Heat and Mass Transfer 48 (2005) 4640–4651.
- [9] Akhavan-Behabadi M.A, Kumar R, Salimpour M.R, and Azimi R, Pressure drop and heat transfer augmentation due to coiled wire inserts during laminar flow of oil inside a horizontal tube, International Journal of Thermal Sciences 49 (2010) 373–379.
- [10] Munoz-Esparza D, and Sanmiguel-Rojas E, Numerical simulations of the laminar flow in pipes with wire coil inserts, Computers & Fluids 44 (2011) 169–177.
- [11] Versteeg H. and Malasekara W, An Introduction to Computational Fluid Dynamics, (2nd edition, year of publication-2007), Pearson Publications.